CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

In this chapter, it will discuss about the process that will go through to make the excellent outcome and to fulfill the objective of research. Generally, this research will focus on the behavior and probabilistic analysis of steel jacket structure using finite element method. Methodology is a guideline for the process in this project. The project also can run smooth way using this approach.

Fix jacket frame also will be model in 3D and analyzed to obtain the results. Here, the model will be prepared with steel properties, assuming the size by using ANSYS software. The analysis of steel jacket frame consist 3 phases. Firstly, preprocessing phase – defining the problem, model and the structure. Secondly, solution phase – assigning all the load, constraints and solve the analysis of structure. Lastly, Post-processing phase – viewing the results.

The design of steel jacket structure is formulation of a plan to give an engineer build a good product with a specified performance objective. This process consist a few of steps, and the parts of the process may need to be repeated many times before production of a final product can begin. The design process starts with preprocessing and ends with getting the moment, stress and strain for the model.

For a clear understanding of this chapter, a flow chart has been drawn to show the process of research.

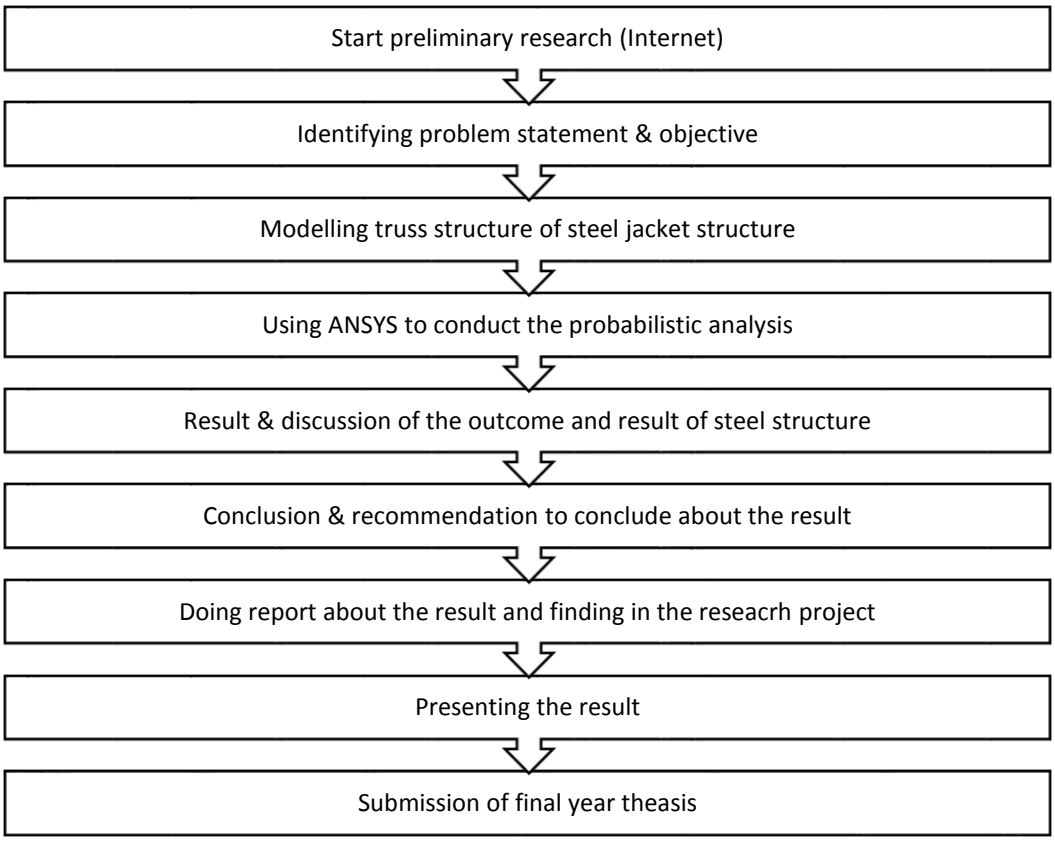

Figure 3.0: Flow Chart of The Research Process

3.2 PREPROCESSING: DEFINING THE PROBLEM

In this processing level, we define problem for frame jacket structure (offshore structure).

3.2.1 Modelling 3D Jacket platform structure

ANSYS software is used to create this 3D steel frame jacket model. For the information, this structure is about of offshore structure. First of all, the coordinate for each point of structure is to be determined and insert as key points in the software to create the shape of the model of frame structure in 3D form. After that, the key points are to be joined together to form the frame structure shape in 3D. Finally, this model will be analyzed to get the result of deflection, stress and strain.

3.2.2 Entering Title

Firstly, a title for the project shall be named by following the commands. Select "File" from the Utility Menu and click "Change Title". Create a title for the project as "STEEL FRAME JACKET PLATFORM". After that, the title will come out at the bottom left of window. This step shown in Figure 3.1.

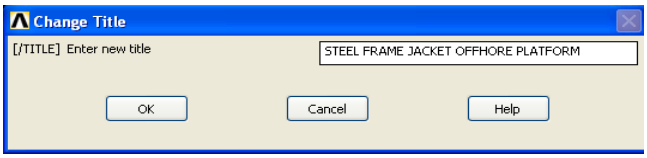

Figure 3.1: Change Title

3.2.3 Defining CivilFEM

ANSYS CivilFEM must be active because CivilFEM will anaized the jacket structure with come out the result such as deflection, stress and strain. On the left side screen, we can see the ANSYS Main Menu, then we should click CivilFEM and then the activate CivilFEM like Figure 3.2 will come out from window. To activate CivilFEM, we just only Click ' OK '.

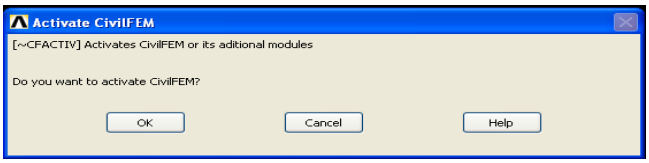

Figure 3.2: Activate CivilFEM

After that, choose the type of code checking and designing in CivilFEM setup option and also choose the International System SI unit. In the ANSYS main menu, click "Civil Setup" and CivilFEM Setup Option will come out on the window. They have a few type code checking such as Eurocode 3 for steel code, Eurocode 2 for reinforced concrete code and Pre-stressed concrete code, Eurocode 8 for Seismic code. Click "steel code", select "Eurocode 3" then click "OK" for design this project. This step shown in Figure 3.3 Activate CivilFEM.